Truck Aerodynamics:

Large-Eddy Simulation (LES) using the Finite-Element Method (FEM)

Rose McCallen, Ph.D.
Lawrence Livermore National Laboratory

March 1999

University of California
Lawrence Livermore National Laboratory

We need advanced CFD tools to accurately predict drag effects for trucks.

Flow is time dependent, 3D with a wide range of scales

- Flapping recirculation zones
- Thin boundary layers transition and separate
- Flow tripped by head lamps, grab handles, etc.
- Everything upstream effects what happens downstream

To reduce experimentation, accurate CFD with less empiricism is needed

- Commercial CFD tools do not predict correct drag effects - per Industry
- Models have adjustable empirical parameters

Chosen approach

- Large-eddy simulation (LES) using the finite element method (FEM)

LES is a challenge but we have the experience and resources to succeed.

Outline

Background

Approach

Challenges

Plan and Progress

Formulation and implementation

Benchmarking

Truck simulation

Current Technology

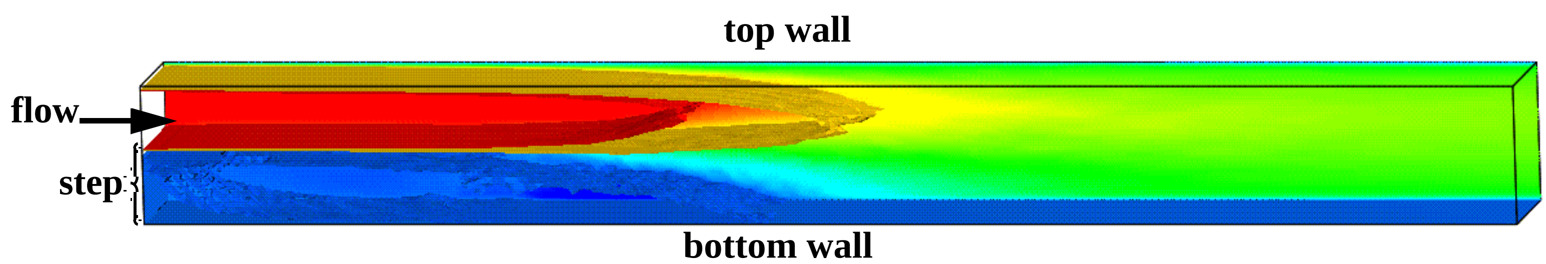
RANS/LES hybrid - paper review

Large-eddy simulation provides a wealth of information and less empiricism.

Reynolds-averaged Navier-Stokes (RANS)

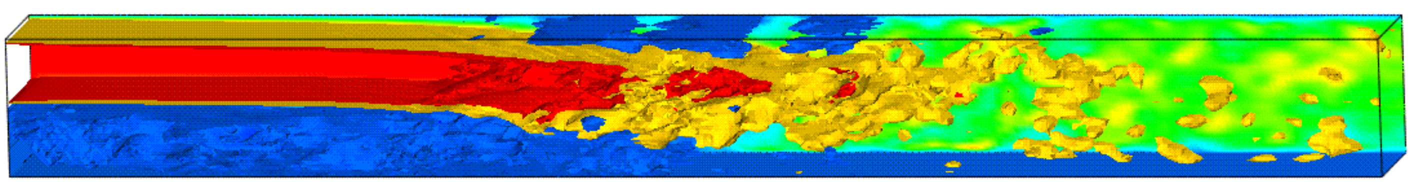
- Many empirical parameters
- Time-averaged solution

Backward-facing step: streamwise velocity



Large-eddy simulation (LES)

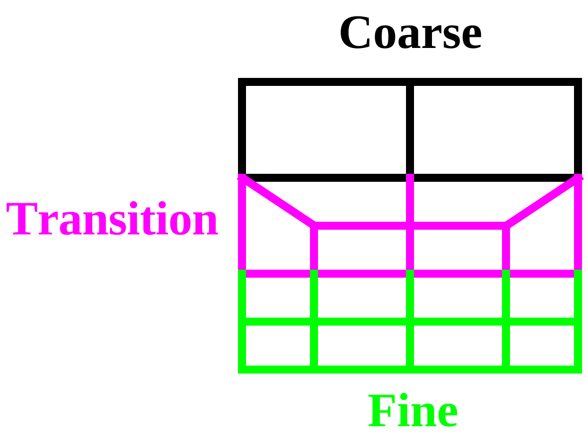
- One empirical parameter -> less empiricism
- 3D, unsteady solution of vortex shedding

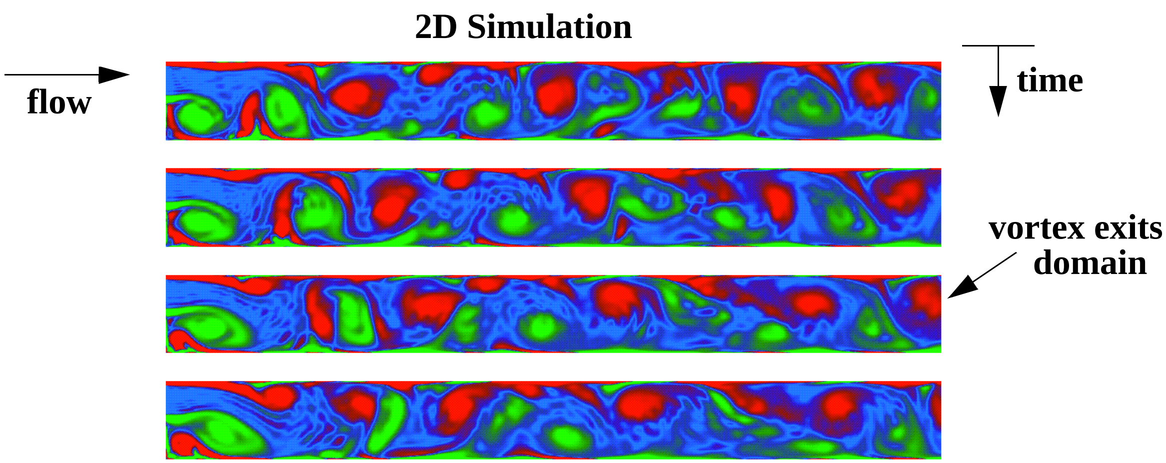


LES/FEM provides a unique approach for solving practical problems.

Advantages of FEM

- **Unstructured meshes**
- **Natural boundary conditions**
- **Coupling to other FEM codes**





Zero natural boundary conditions capture the vortical outflow

The LES challenges are related to physical as well as numerical modeling.

Boundary layers are too thin	Can't resolve boundary layers - problem gets too big
	Wall approximations in development
Runtime too long	Evolution is over long time scales
	Parallel computations/solvers required - in development
Analysis	Huge data sets
	Visualization required - in development
	Methods for testing convergence (V&V) in development

Significant development being done by LLNL programs.

The first year deliverable is to ‘integrate’ and develop the flow model and complete a demonstration problem.

Milestone	**FY99 incompressible flow demonstration**
R&D	**Solver integration/parallelization**
	Turbulence modeling
	Boundary conditions
	Data analysis
Approach	**Utilize existing methods, tools, resources, etc.**
	- Existing/tried formulation
	- Smagorinsky SGS model for FY99
	- Integrating existing codes
	Take advantage of the Lab’s infrastructure

keep it simple

For the incompressible flow modeling we are taking advantage of existing methods and codes.

ALE3D
structural/thermal/chemistry/compressible-flow

Incompressible LES/FEM Code

Tasks	Status
Establish compatibility and flexibility of the formulation	✓
Extract physics coding from existing code and modify for ALE3D	✓
Establish parallelization approach and implement	in progress
Coding for input/output and postprocessing issues	in progress
Benchmark testing	

The formulation is an established method, but solver implementation ‘was’ an issue.

Formulation, solution approach, and coupling

$$\boxed{(C^T M^{-1} C)P^n = C^T A^n}$$

Solve for pressure

where $A^n = M^{-1}[(K^n + N(u^n))\underline{u}^n - F(u^n)]$

$$\underline{u}^{n+1} = \underline{u}^n - \delta t(A^n - M^{-1}CP^n)$$

Update velocity

where

$$C = \begin{bmatrix} c_{in(1)} \\ c_{in(2)} \\ c_{in(3)} \end{bmatrix} ; c_{in(\alpha)} = \int_\Omega \psi_n \frac{\partial \varphi_i}{\partial x_\alpha} \quad M = \begin{bmatrix} m_{ij} & 0 & 0 \\ 0 & m_{ij} & 0 \\ 0 & 0 & m_{ij} \end{bmatrix} ; m_{ij} = \int_\Omega \varphi_i \varphi_j$$

$C^T M^{-1} C$ is only a function of geometry

Matrix characteristics and solver

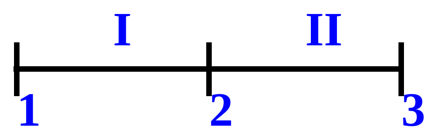
Piecewise constant pressure basis functions (‘zone centered’)

Setting-up a row at a time!

Can’t use solver’s Finite Element Interface (FEI)

The $C^T M^{-1} C$ matrix is *global* and can't be constructed element-by-element.

One-Dimensional Example



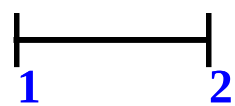
$$(C^T M^{-1} C)P = rhs$$

$$\begin{bmatrix} c_{1I} & c_{2I} & 0 \\ 0 & c_{2II} & c_{3II} \end{bmatrix} \begin{bmatrix} m_{1I} & 0 & 0 \\ 0 & (m_{2I} + m_{2II}) & 0 \\ 0 & 0 & m_{3II} \end{bmatrix} \begin{bmatrix} c_{1I} & 0 \\ c_{2I} & c_{2II} \\ 0 & c_{3II} \end{bmatrix} \begin{bmatrix} P_I \\ P_{II} \end{bmatrix} = rhs$$

$$\begin{bmatrix} c_{1I} m_{1I} c_{1I} + c_{2I}(m_{1I} + m_{2II}) c_{2I} & \boxed{c_{2I}(m_{2I} + m_{2II}) c_{2II}} \\ \boxed{c_{2I}(m_{2I} + m_{2II}) c_{2II}} & c_{2II}(m_{2I} + m_{2II}) c_{2II} + c_{3II} m_{3II} c_{3II} \end{bmatrix} \begin{bmatrix} P_I \\ P_{II} \end{bmatrix} = rhs$$

But, element-by-element formation looses the off-diagonal terms

$$\begin{bmatrix} c_1 & c_2 \end{bmatrix} \begin{bmatrix} m_1 & 0 \\ 0 & m_2 \end{bmatrix} \begin{bmatrix} c_1 \\ c_2 \end{bmatrix} \begin{bmatrix} P \end{bmatrix} = rhs$$

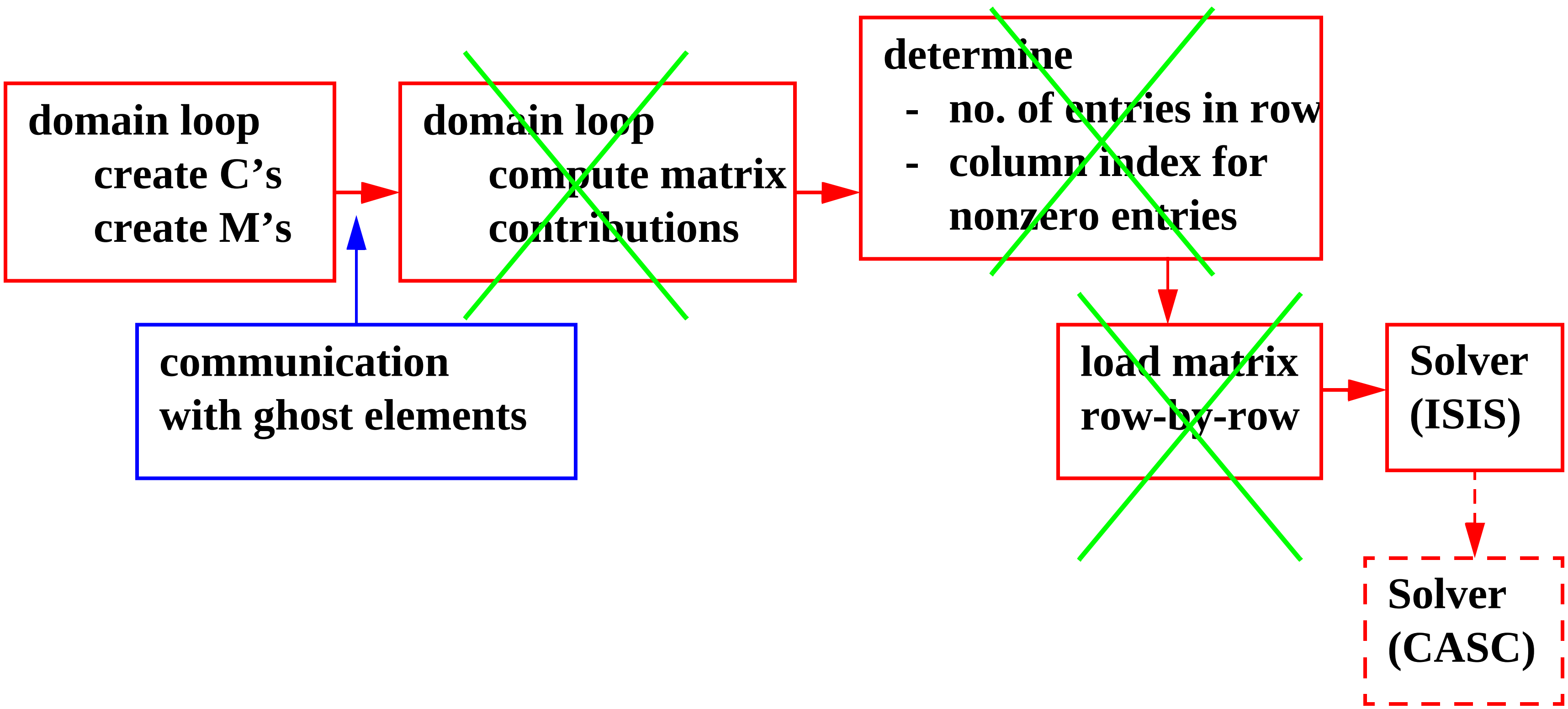


$$(c_1 m_1 c_1 + c_2 m_2 c_2)P = rhs$$

The matrix multiply must be done globally.

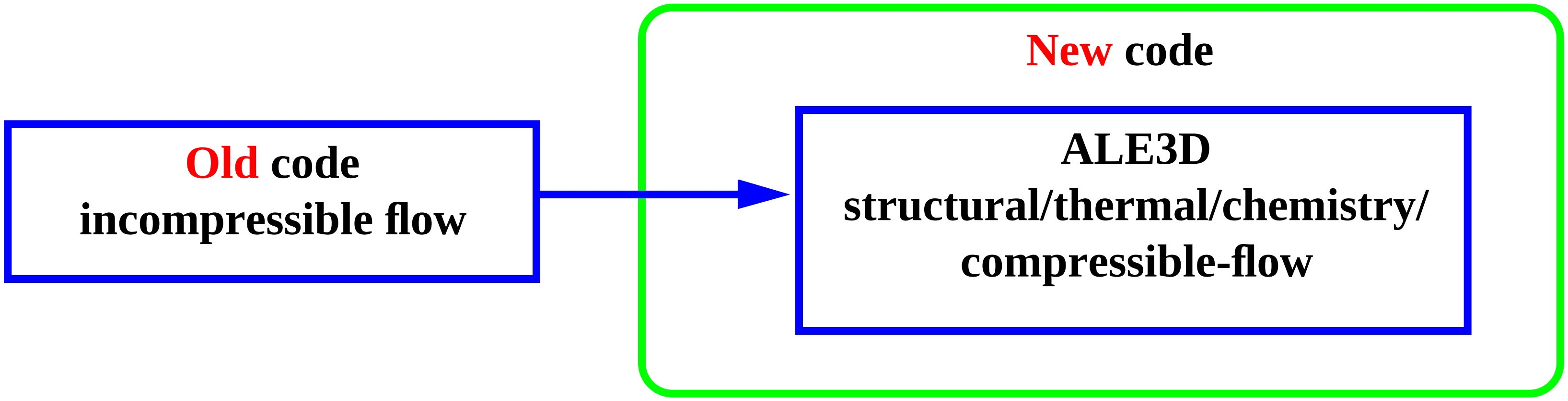
If the FEI performs the matrix multiply, the parallelization effort is significantly reduced.

Parallelization outline



A complete but expedient V&V method will be used.

Verification & Validation



For an existing example problem (e.g., backward-facing step)

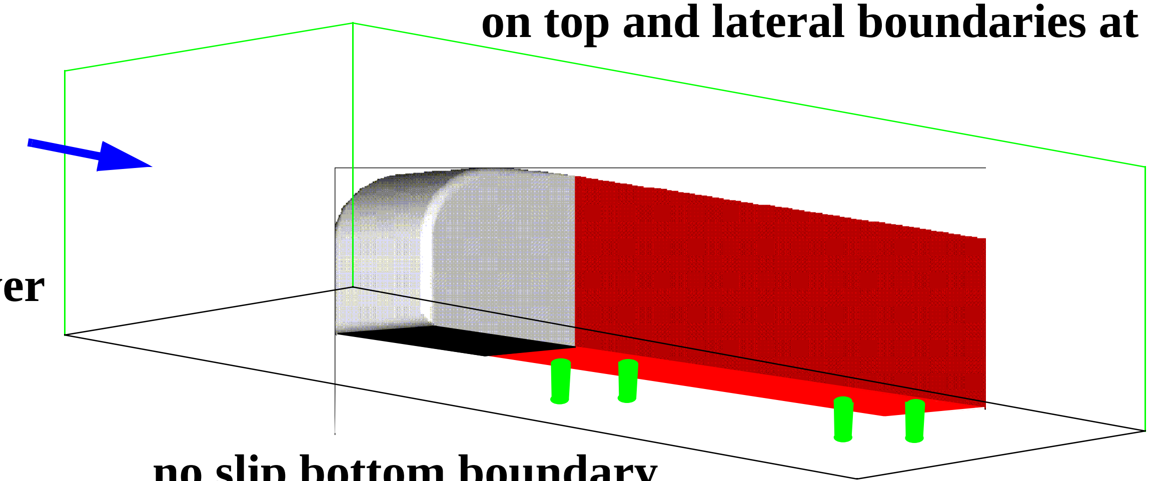
Question	Cases
Do old and new code agree?	compare to serial run with direct solver
Does iterative solve work?	compare serial run with iterative and direct solve
Does code run in parallel?	compare serial vs. parallel run for iterative solve

Simulating the NASA 7’x10’ wind tunnel experiment is the demonstration problem.

Domain and boundary conditions are chosen to minimize grid size

slip with no penetration
on top and lateral boundaries at tunnel walls

flow inlet
far enough
upstream to
capture
boundary layer



zero natural
boundary
condition at
outflow

no slip bottom boundary
and truck walls
using Smagorinsky SGS model to wall

Compressible as well as incompressible simulations can be performed with an unstructured grid.

Plan

Compressible (Ma > .1)

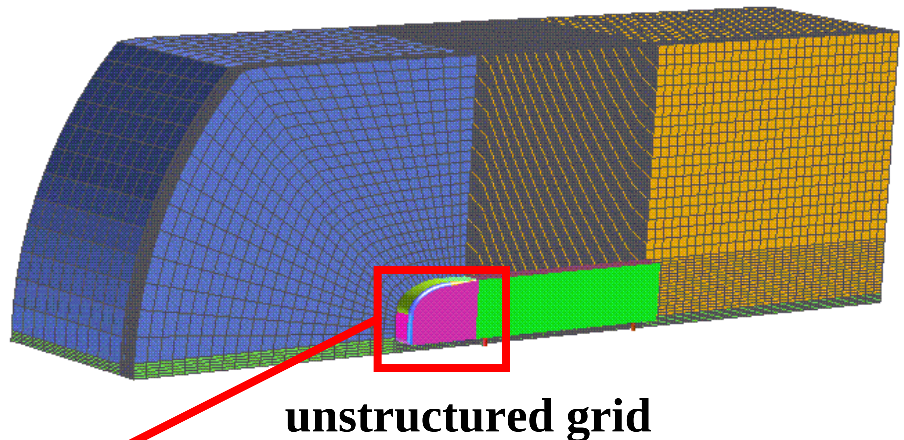
NASA 7’x10’ results for Ma = 0.27

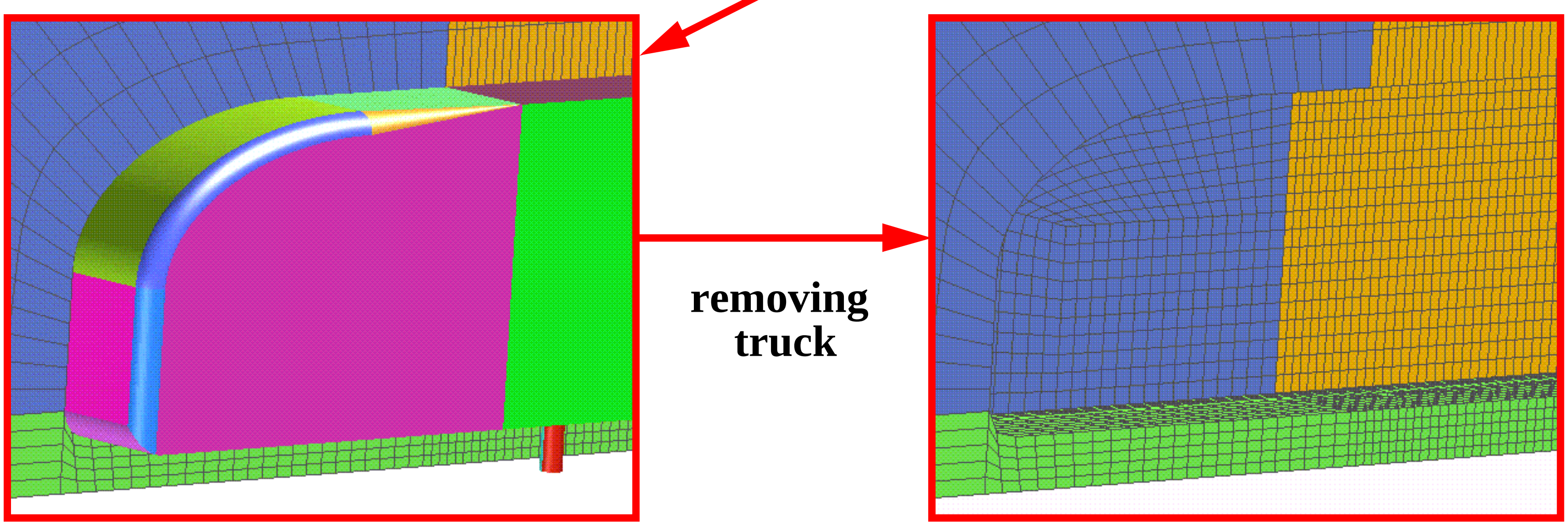
Incompressible (Ma < .1)

NASA 7’x10’ results for Ma < 0.1

USC results for 200,000 < Re < 400,000

Texas A&M for Re = 1,600,000





Paper Review: LES is not feasible for attached flow (wings), but desirable for separating flow.

Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach

P.R. Spalart, W-H. Jou, M. Strelets, S.R. Allmaras, 1998.

Detached-Eddy Simulation (DES) method looks promising

DES offers RANS in the boundary layers and LES after massive separation, within a single formulation.

For thin shear flows the grid resolution exceeds current capabilities (e.g., front/sides of cab)

... (for a wing) we find the need for 10^{11} grid points, under the most favorable conditions. Today, a calculation with 10^{8} points is impressive.

Coarse grids can be used in separation regions (e.g., truck wake)

... the most challenging flow regions for turbulence models "trained" in thin shear flows are the regions of massive separation ... driven by low-aspect-ratio features such as wheels and flap edges. This is where LES is most desirable ... would not require orders of magnitude in grid refinement.

Unstructured and adapted grids are required

... the grid coarsens as soon as possible outside the boundary layer; the irrotational region allows a grid spacing much larger than the boundary-layer eddies.

Much has been done and much needs to be done.

LES/FEM has advantages

- Accuracy with less empiricism
- Built-in outflow conditions and unstructured grids

LES/FEM has challenges

- Wall approximations
- Parallel computations
- Data analysis methods

Approach

- Take advantage of existing methods and codes - keep it simple

Current Technology

- DES method attempts to solve boundary layer resolution problem

LES/FEM is a challenge but we have the experience and resources to succeed.